

FreeCAD-CFD Workbench

Tutorial 3: Boundary layer meshing and turbulent flow



CFD Workbench

WORKBENCH

This workbench aims to help users set up and run CFD analysis. It guides the user in selecting the relevant physics, specifying the material properties, generating a mesh, assigning boundary conditions and setting the solver settings before running the simulation. Where possible best practices are included to improve the stability of the solvers.

INSTALLATION

WINDOWS:

- <https://www.freecadweb.org/wiki/Download>
- Install CfdOF from Tools | Addon manager
- Go to Edit | Preferences | CFD to check and install dependencies

LINUX:

- https://www.freecadweb.org/wiki/Install_on_Unix
- Install CfdOF from Tools | Addon manager
- Install OpenFOAM (<https://openfoam.com/download/>)
- Install Paraview
- Go to Edit | Preferences | CFD to check dependencies and install cfMesh

LATEST INFORMATION

Please see the CfdOF [README file](#) for up-to-date information.

LEAD DEVELOPERS

Johan Heyns (CSIR, 2016-2018) jaheyns@gmail.com,

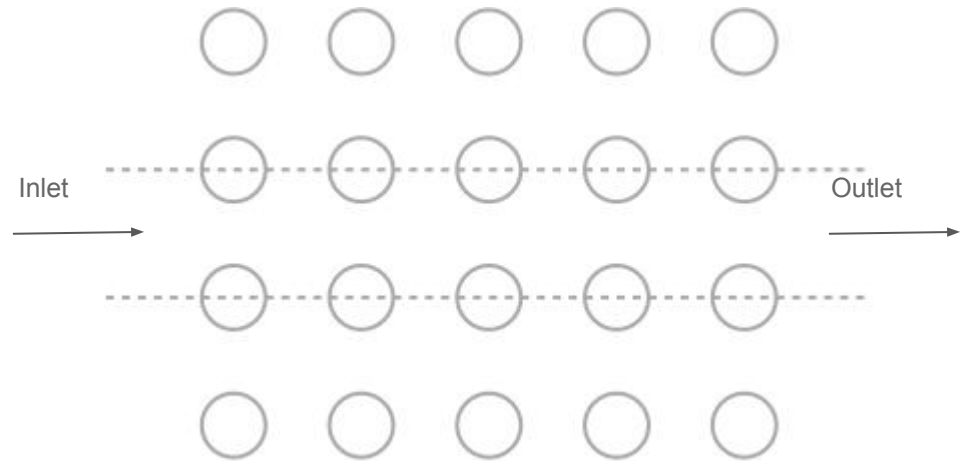
Oliver Oxtoby (CSIR, 2016-2018) oliveroxtooby@gmail.com,

Alfred Bogaers (CSIR, 2016-2018) alfredbogaers@gmail.com,

Part Design

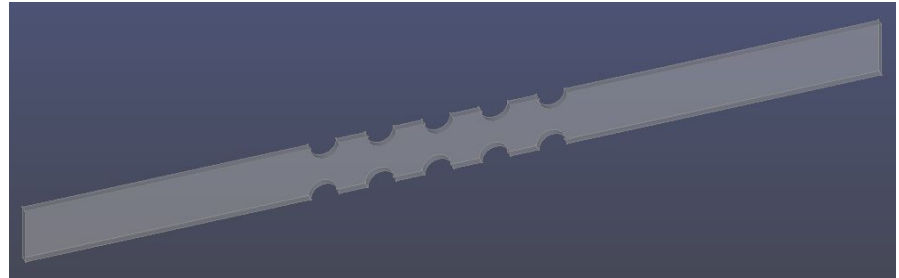
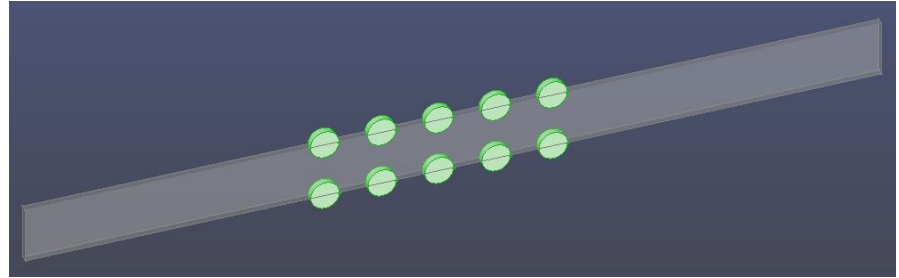
Tube bundle

- To demonstrate how to model viscous flow, the pressure drop over a tube bundle is calculated.
- Bundle parameters
 - Tube diameter: 100 mm
 - Tube pitch: 200 mm
- Assuming symmetric conditions we model the flow through one of the channels.



Create channel

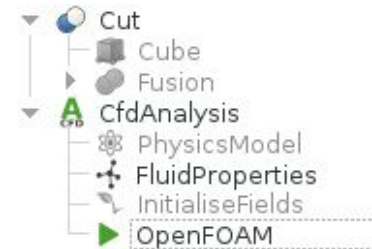
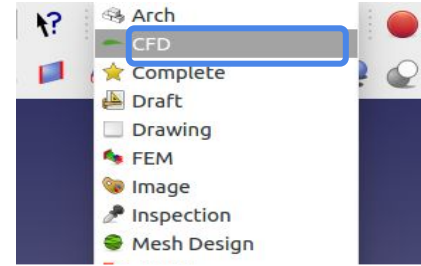
- Activate the “Part” workbench
- Create a cube
 - Length: 3000 mm
 - Width: 200 mm
 - Height: 25 mm
- Create 10 cylinders
 - Diameter: 100 mm
 - Spacing: 200 mm
- For the channel cut the cylinders away from the cube



Boundary layer meshing

Initialise CFD analysis

- Activate the CFD WB by clicking on the dropdown menu in the taskbar and select “CFD”
- Create a CFD analysis which will automatically generate:
 - Physics model
 - Fluid properties
 - Initialise fields
 - OpenFOAM solver

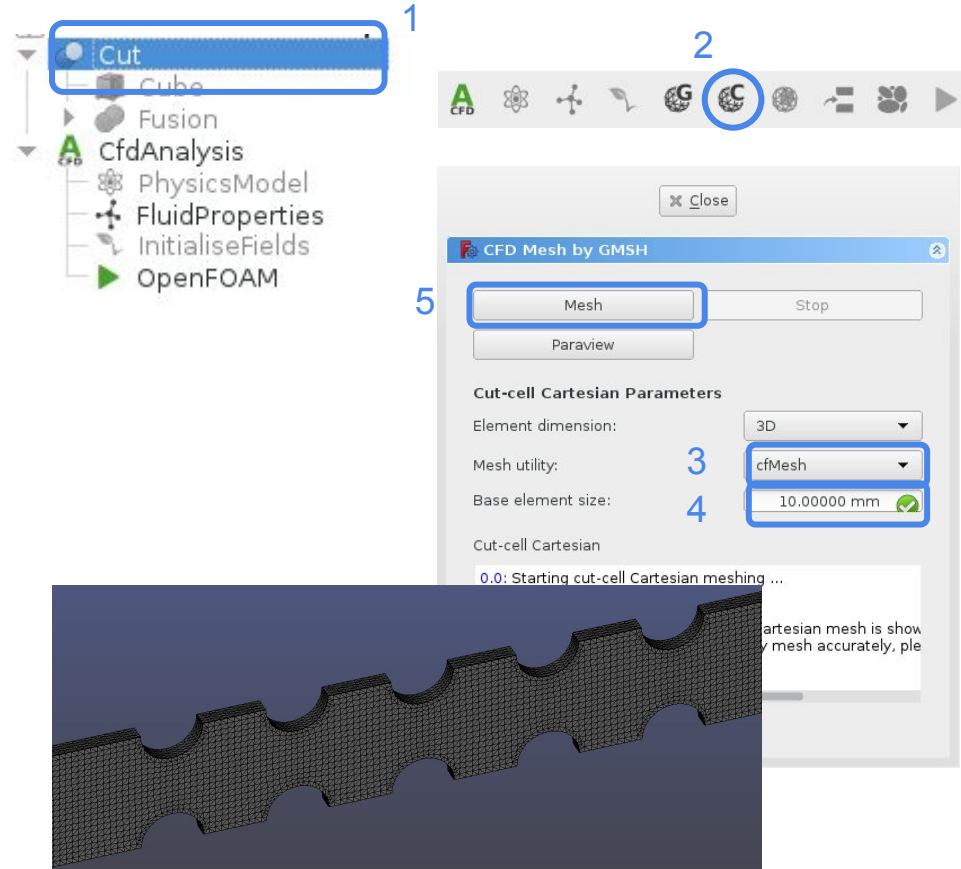


Create an initial mesh


- To create a preliminary mesh select the 'Cut' object.
- Click on the cut-cell Cartesian icon
- Select the 'cfMesh' utility which allows for boundary layer meshing.
- Set the maximum element characteristic length to 10 mm and mesh.
- You should now have a uniformly spaced cut-cell Cartesian mesh.

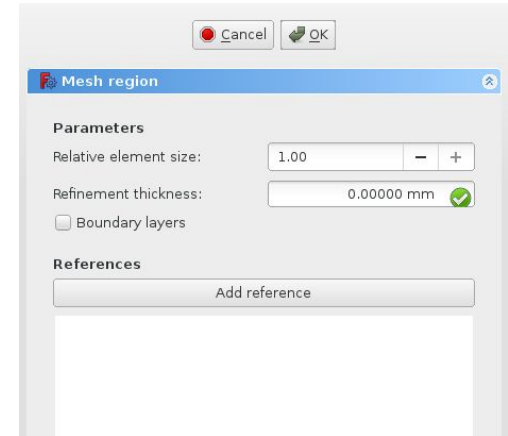
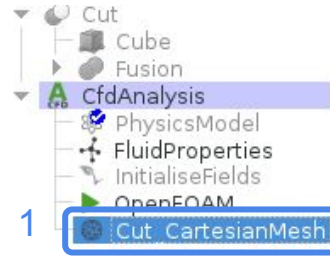
NOTE: Currently, only the 'cfMesh' utility supports boundary layer meshing.

NOTE: See Tutorial 1 for information on how to create a 2D mesh



Adding boundary layers

- Highlight the mesh object to activate the mesh region icon 
- Create a mesh region by clicking on the icon .
- A mesh region object will now have been created (if not visible expand “Cut_CartesianMesh”).
- Double click it to open the task panel.
- In this task panel, we can now edit the refinement and boundary layer parameters.

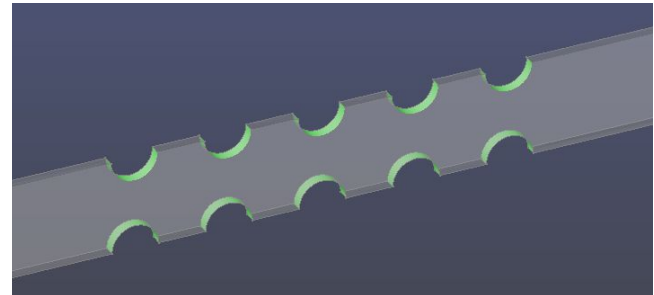


Mesh region parameters

- The refinement parameters ‘Relative element size’ and ‘Thickness’ are respectively used to set the cell size relative to the base mesh and the absolute thickness of the refinement region.
 - Relative element size: 0.5
 - Refinement thickness: 20 mm
- Check the “Boundary Layer” box
- The user can set the number of layers, the expansion ratio that governs the growth of the layers and, optionally, the maximum first cell height (The default 0 mm will be ignored).
 - Number of layers: 15
 - Expansion ratio: 1.2
- Next, the reference faces need to be added.

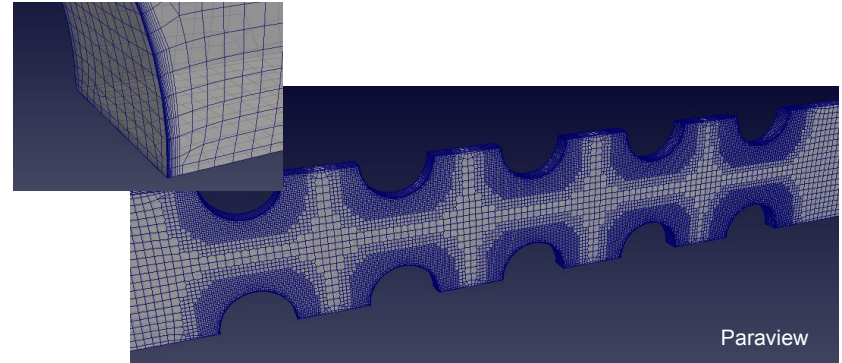
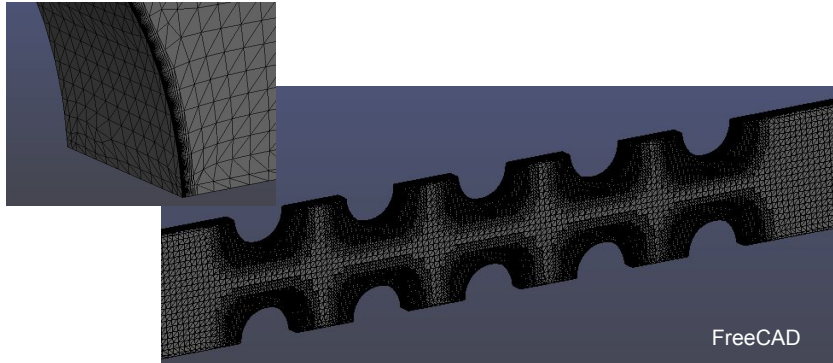
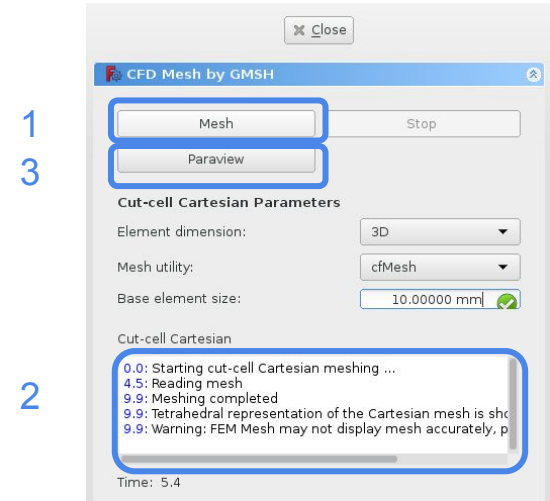
NOTE: The expansion ratio is limited to be greater than 1.0 and less than 1.2.

1
2
3
4



Update mesh

- Once the mesh region parameters are updated the user can go back to the mesh task panel and click on 'Mesh' to recompute.
- The message console shows the progress and once meshing is completed a tetrahedral representation of the mesh is displayed.
- For a more accurate representation of the mesh click on "Paraview" to open the viewer.



Viscous analysis

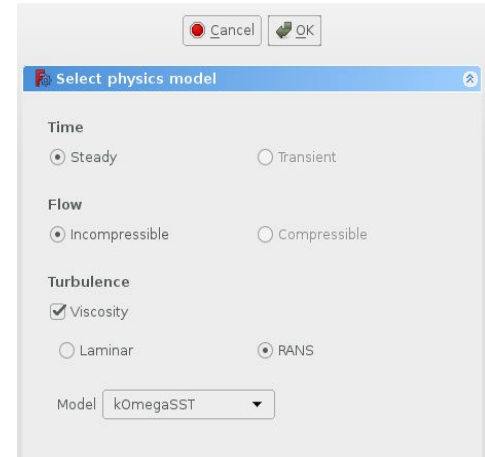
RANS analysis

- In the “PhysicsModel” task panel under “Turbulence”, select the RANS checkbox which will automatically select the k-w SST model.

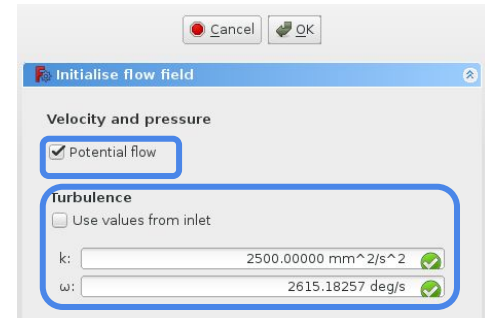
NOTE: Currently the CFD WB only supports the k-w SST model.

- Activate the “InitialiseFields” task panel and ensure the “Potential flow” checkbox is selected. The high aspect ratio boundary layer cells require a well initialised velocity field.
- The k and w can be initialised using the values from the inlet or specified by the user.

1



2



3

Boundary conditions

- In addition to entering the velocity at the inlet boundary, the user is required to specify the turbulence quantities.
- The user can either specify the Turbulence intensity and Length scale or the Kinetic energy (k) and Dissipation rate (w).

NOTE: The following links might be useful in guiding the user in specifying the inlet turbulent quantities:

<https://turbmodels.larc.nasa.gov/sst.html>

https://www.cfd-online.com/Wiki/Turbulence_intensity

https://www.cfd-online.com/Wiki/Turbulence_length_scale

- For the test problem, we will use turbulent intensity
 - Turbulent intensity: 5 %
 - Length scale: 2 mm



Turbulence specification

Intensity & Length Scale ▼

turbulence intensity and eddy length scale

Turbulence intensity (I) 0.05000 ✓

Length scale (l) 2.00000 mm ✓

Turbulence specification

Kinetic Energy & Specific Dissipation Rate ▼

k and omega specified

Turbulent kinetic energy (k) 1080.00000 mm²/s² ✓

Specific dissipation rate (ω) 57.29578 deg/s ✓

Boundary conditions

- Viscous/no-slip boundaries should be applied to the tube walls.
- For no-slip walls, wall functions are used for the turbulent quantities. This requires the y^+ of the first cell to be between 30 and 150.

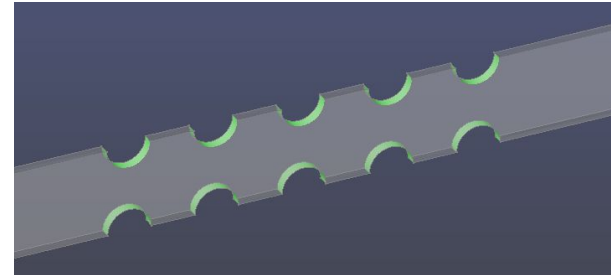
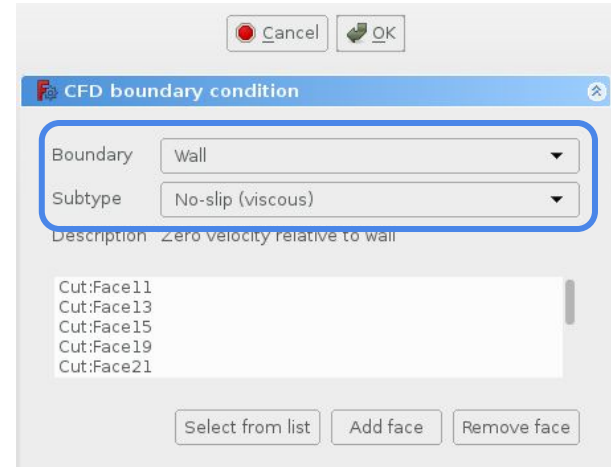
NOTE: The following links might be useful in guiding the user to compute the non-dimensional y^+ value:

[https://www.cfd-online.com/Wiki/Dimensionless_wall_distance_\(y_plus\)](https://www.cfd-online.com/Wiki/Dimensionless_wall_distance_(y_plus))

https://www.cfd-online.com/Wiki/Law_of_the_wall

- The remainder of the walls may be treated as either “Slip” or “Symmetric” and “Static pressure” is prescribed at the outlet.

NOTE: An undefined boundary patch will default to slip/inviscid wall.



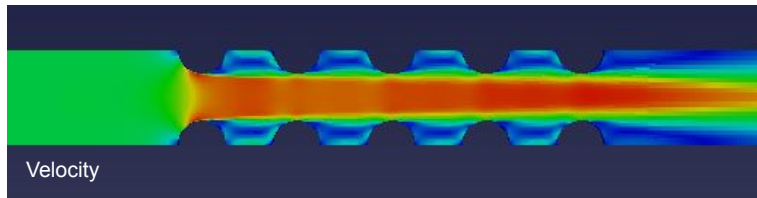
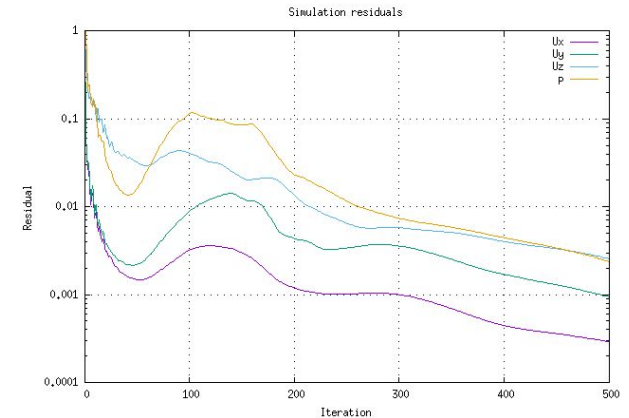
Running the simulation

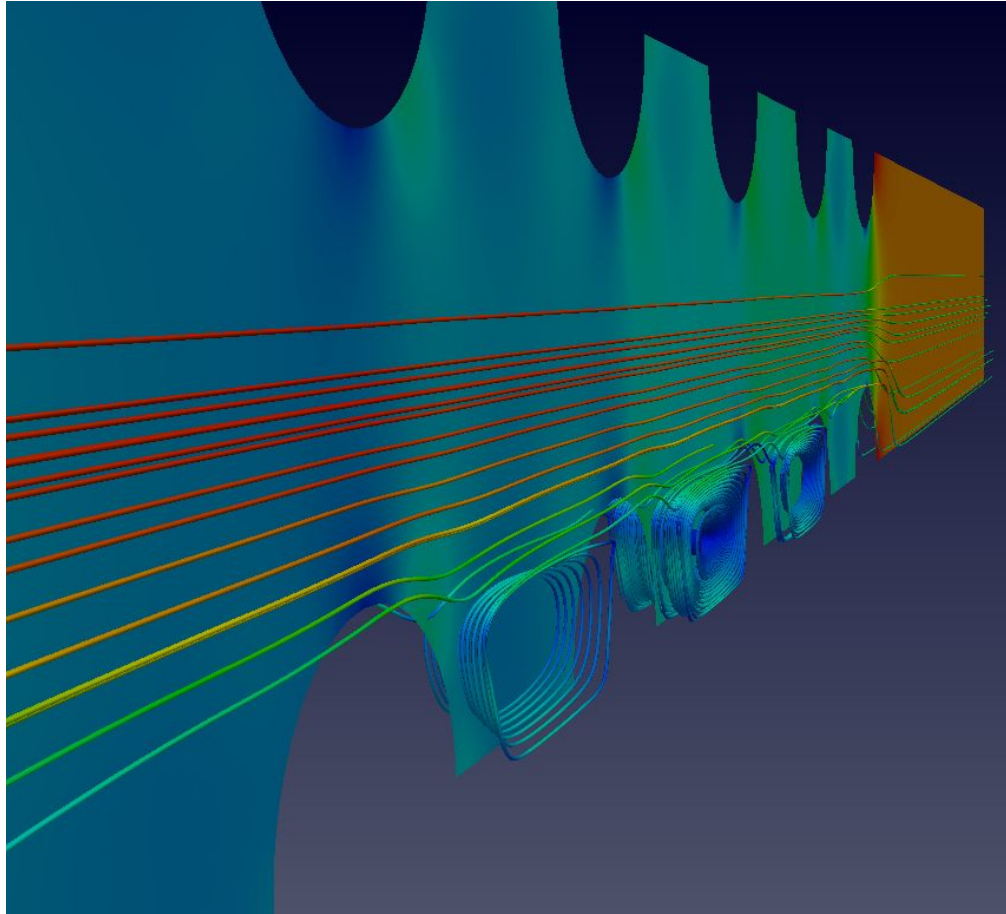
Simulation

- In the “CfdSolver” data panel, the parallel setting can be set to true to run over multiple cores.
- The solver Task panel may then be activated, from which the case is written and simulation run.
- The residual plot is automatically created and the results can be viewed in Paraview.

NOTE: The incompressible solver in OpenFOAM stores the kinematic pressure, $p = P/\text{density}$.

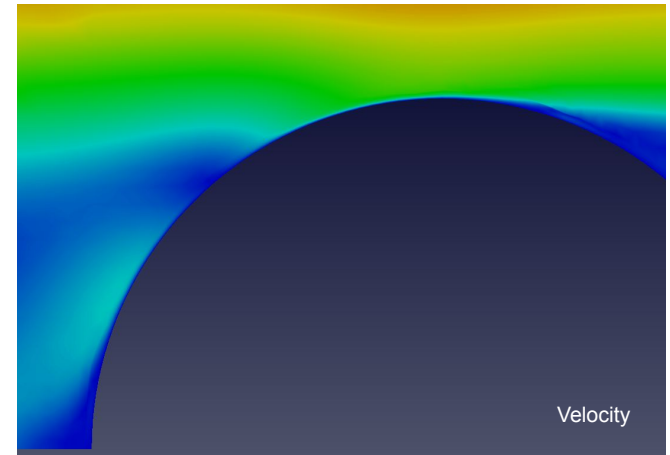
Solver	
Input Cas...	case
Parallel	true
Parallel C...	4
Solver N...	OpenFOAM
Working Dir	/tmp
Time Ste...	
Converg...	0.00010
End Time	500.00000
Time Step	1.00000
Write Inte...	10.00000





Paraview:

- “Slice” to show the pressure contours.
- “StreamTracer” to show the streamlines
- “Tube” to apply thickness to the streamlines



The End